Defense Technical Information Center Compilation Part Notice

ADP023096

TITLE: Validation of Helicopter Flow Predictions Using Wind Tunnel LDA Measurements

DISTRIBUTION: Approved for public release, distribution unlimited

This paper is part of the following report:

TITLE: Proceedings of the Ground Target Modeling and Validation Conference [13th] Held in Houghton, MI on 5-8 August 2002

To order the complete compilation report, use: ADA459530

The component part is provided here to allow users access to individually authored sections of proceedings, annals, symposia, etc. However, the component should be considered within the context of the overall compilation report and not as a stand-alone technical report.

The following component part numbers comprise the compilation report:
ADP023075 thru ADP023108

Validation of Helicopter Flow Predictions Using Wind Tunnel LDA measurements

David Gatland, Dr. Lee Hayward, Dr Steve Rooks
Dstl, Air Systems,
A2 Bldg, 1064,
Farnborough, Hampshire
GU14 0LX

ABSTRACT

Helicopter engine exhaust plumes can constitute a large contribution to the total platform infra-red (IR) signature. The flow around such vehicles is particularly complex and an accurate representation of the plume is essential for high fidelity prediction of the IR signature. CFD techniques are well documented and have long been used for the generation of airflow and thermodynamic data. However. validation is essential for any problem to which the techniques are applied. Dstl uses the PHOENICS CFD code for the modelling of aircraft flow fields. In conjunction with PHOENICS, the SAPPHIRE modelling suite has been developed to aid model generation and IR signature analysis of air platforms.

This paper reports on the ability of SAPPHIRE to accurately predict the interaction of an exhaust plume with downwash caused by a helicopter rotor. Laser Doppler Anemometry (LDA) has been used to measure the flow around a simplified wind tunnel model orientated such that the free stream simulates the downwash. Comparison of this data with CFD results shows good general correlation and gives confidence that a 'fit for purpose' solution can be generated. Possible reasons for the discrepancies have been identified and are discussed.

INTRODUCTION

At Dstl Farnborough, the Air Systems Department includes a team concerned with the measurement, prediction and assessment of air-vehicle infra-red (IR) signatures. The SAPPHIRE suite of software has been developed to aid the generation and solution of flow models and IR signature predictions for various platforms. The on-going process of development and validation of SAPPHIRE has been reported regularly over recent years at the GTM&V conference [1,2,3,4].

The flow field relating to rotary wing aircraft is particularly complex and involves the interaction of the free stream flow, rotor downwash, tail rotor flow, engine intake flow and engine exhaust. In addition to these main features, the flow can also be affected by the proximity to the ground and the numerous inlets and outlets for items such as IR suppressors. Figure 1 shows the main features described above.

A key characteristic of the flow is the interaction between the rotor downwash and the engine exhaust, as this determines the position and size of the exhaust plume as well as potential hot flow impingement on the fuselage/tail boom. Accurate modelling of this interaction is important for any signature prediction and potential evaluation of IR exhaust suppressors. The SAPPHIRE helicopter models have been developed to account for such complex issues including the rotor flow field interaction. [1]

Validation of SAPPHIRE to model individual flow cases is an important and ongoing process. Well controlled wind tunnel tests are considered to be the main source for validation data. However, validation experiments for helicopters are few in number [5]. The main reason for this shortfall is the complexity inherent in measuring a 'real' helicopter flow field. Detailed flow field measurements are virtually impossible to make while a helicopter is in flight, while ground static measurements are rendered invalid due to the interaction of the rotor downwash with the ground. Therefore the process for validation of a code normally involves breaking down the problem into simple 'building block' experiments which look at the main factors individually.

In the mid 1990s, in support to helicopter infra-red suppressor assessments, a rotor downwash study jointly funded by UK MOD and Rolls Royce provided high fidelity flow field data for a generic helicopter/exhaust model. This study was specifically configured to look at the interaction of exhaust flows with rotor downwash.

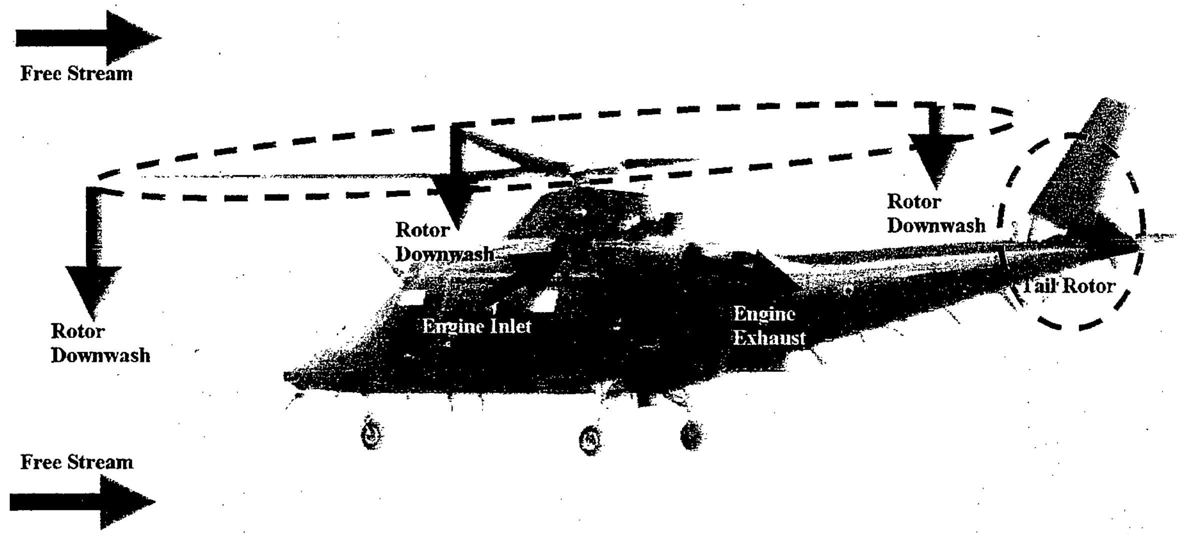


Figure 1: The Complex Flow Field Surrounding Helicopter Platforms.

The data from this study has recently been identified as a useful source for comparison with a SAPPHIRE helicopter flow field prediction. A CFD representation of the configuration has been produced and CFD predictions compared with the measured data. This paper reviews the numerical analysis and the related study carried out.

The aim of this study was to provide confidence that SAPPHIRE can predict the main flow characteristics resulting from the interaction of an exhaust flow with downwash. To be 'fit for purpose', the prediction needs as a minimum to show the general trends and position of the flow, along with the mixing and attenuation of the plume.

EXPERIMENTAL/COMPUTATIONAL METHODOLOGY AND TECHNIQUE

Wind Tunnel / LDA Setup

The Westland Helicopters Ltd (WHL) 12ft x 10ft low speed wind tunnel was used for the rotor downwash study. A generic helicopter model was mounted within the tunnel and orientated normal to the onset flow to simulate the rotor downwash. A separate flow was pumped through the model to simulate the engine exhaust. Figure 2 shows the exhaust and tail boom sections of the model mounted in the wind tunnel.

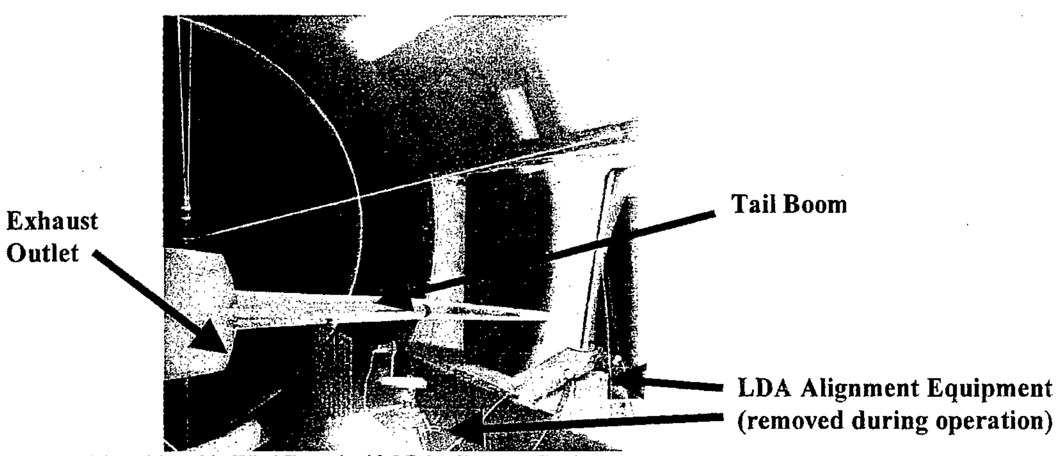


Figure 2: Helicopter Model Positioned in Wind Tunnel with LDA Alignment Equipment.

Laser Doppler Anemometry (LDA) was used to measure the flow velocity in four planes aft of the exhaust outlet. These planes were at the exhaust exit plane (y = 0)[1], 150 mm behind the outlet (y = 150), 300mm behind the outlet (y = 300) and 450mm behind the outlet (y = 450). A schematic is shown in figure 3. LDA requires the flow to be seeded with particles (typically smoke particles) and determines the velocity from the doppler shift undergone by the scattered light from six lasers. The main advantage of LDA over other methods of flow measurement (such as pitot/static probes) is the non-intrusive nature of the equipment with only the laser beams situated in the working section. This method does not require calibration and once the beams are aligned is highly resistant to experimental drift.

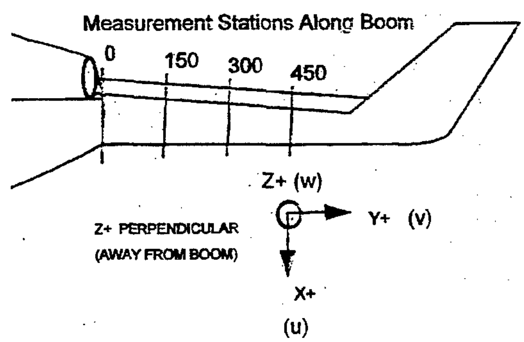


Figure 3: LDA Measurement Planes and Axis Definition.

SAPPHIRE/CFD model

a) General

For the CFD solution a mono-block fully structured grid was created from a CAD model using the FEMGEN module of SAPPHIRE as described in reference [1]. Generally, for problems such as this, an unstructured or multi-block CFD meshing technique and appropriate solver would be the preferred choice. Such techniques enable increased grid densities to be applied in areas of flow complexity or specific interest. Dstl has the ability to use such techniques with solvers such as FLUENT. However, at present these techniques have not been incorporated into the SAPPHIRE suite.

For a structured grid, memory constraints and computing power can be the major factor limiting the size and resolution of the grid. The CFD model is therefore confined to the port side of the geometry with a symmetry plane positioned along the vertical axis of the helicopter. The first stage of the investigation involved the creation of a CFD model with a relatively coarse grid. The aim of this was to generate a 'first pass' result that would indicate SAPPHIRE's general ability to model this case and highlight any potential problems without incurring the time penalty required for a higher resolution grid.

Once this stage had been completed, the geometry was adapted and the grid density increased significantly. Based on the coarse grid model, the flow around the forward section of the helicopter model was deemed to have no significant effect on the solution as the model represented an aircraft in hover. Therefore, in order to reduce the memory requirements and avoid cells with excessively high aspect ratio, the forward region of the original model was removed. The final grid (rear half) had 383104 computational cells compared to 66150 in the full preliminary model.

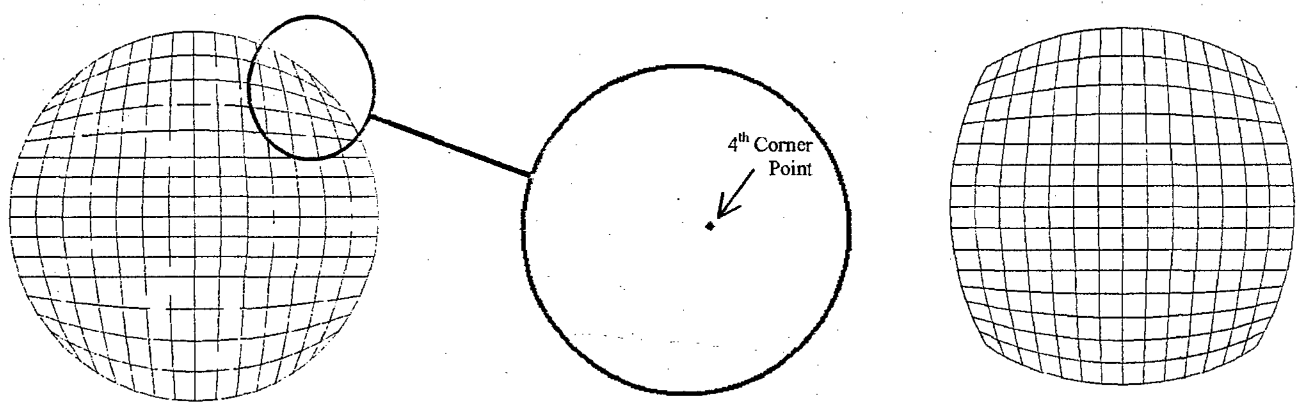
b) Issues

A number of turbulence models are available within the SAPPHIRE suite, including; fixed viscosity, k-W, standard K-ε, modified K-ε, Reynolds stress and renormalization group method (reference [6]). The standard K-ε turbulence model was used during the investigation as it has been validated for a wide range of flow cases and experience has shown it to give good results in this type of flow problem (reference [1]).

---

[1] The LDA data set at y = 0 is not the true exhaust exit plane but an XZ survey plane just aft of the nozzle exit. The true exit plane normal is angled away from the axial (Y) direction.

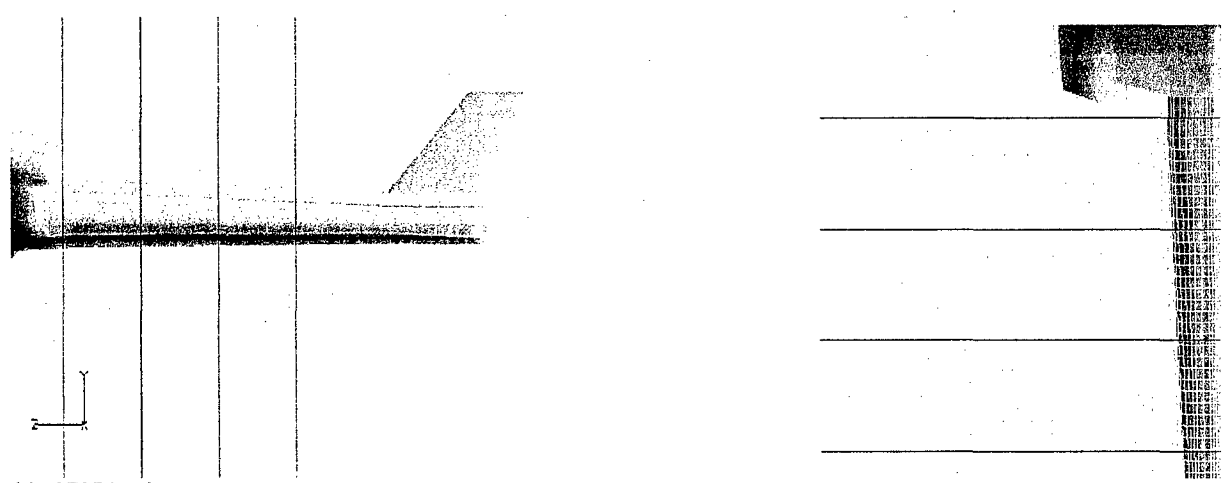
The 'body fitted' mesh is created by manipulating the 8 nodes of each hexahedral cell to give a good approximation to the original CAD geometry. Highly skewed cells and cells of high aspect ratio can cause the CFD solution to become unstable and diverge. Such cells are difficult to avoid completely due to the nature of a 'body fitted' mesh and the necessity of limiting the maximum resolution to the areas of interest only. As the grid density was increased, it was found that the turbulence model, which is less tolerant of these factors than the general flow equations, could not cope with the highly skewed cells generated at the 'corner' points of the circular exhaust geometry (figure 4a). For this case an approximation to the exhaust geometry was made as shown in figure 4b. The tail boom geometry also had to be modified in this way to enable the inclusion of the turbulence model with the fine grid.



a) Round Geometry b) Modified Geometry

Figure 4: Exhaust Outlet Plane Exhibiting a Highly Skewed Cell.

Figures 5a and 5b show cross sections of the mesh and the measurement planes for comparison with the LDA data. The solid region is also shown. For this case, the only region of significant interest is that immediately surrounding, and aft of, the exhaust. The grid was created such that its resolution and quality was greatest in this region.



(a) XY Direction (b) YZ Direction

Figure 5: Picture of CFD Solid Body, Grid Cross Section and LDA Measurement Planes.

RESULTS

Figure 6 shows the U, V and W Cartesian velocity components (see figure 3) given by the coarse CFD result, the LDA data and the fine CFD result at y=150. The 'squaring off' of the fuselage region in the fine grid model discussed above can be seen in figures 6c, 6f and 6i. The main features of the flow field at this point are marked on the LDA plot.

The flow in this plane exhibits a region of high downward velocity (region 1), probably due to the 'venturi effect' as the downwash is forced between the strong exhaust core and the tail boom. This feature can be seen in both CFD results. However, the definition is very poor with the coarse grid. The regions of greatest velocity are fairly similar for the fine grid result and the LDA data, with the peak region predicted further from the tail boom in the CFD model.

The slight recirculation of flow under the exhaust core region in the LDA result (region 2) is virtually unseen with the coarse model but is easily visible in the fine grid result. The accelerated region over, and to the left of the exhaust core (region 3) can be seen in both CFD predictions but is much more accurate in definition with the fine grid.

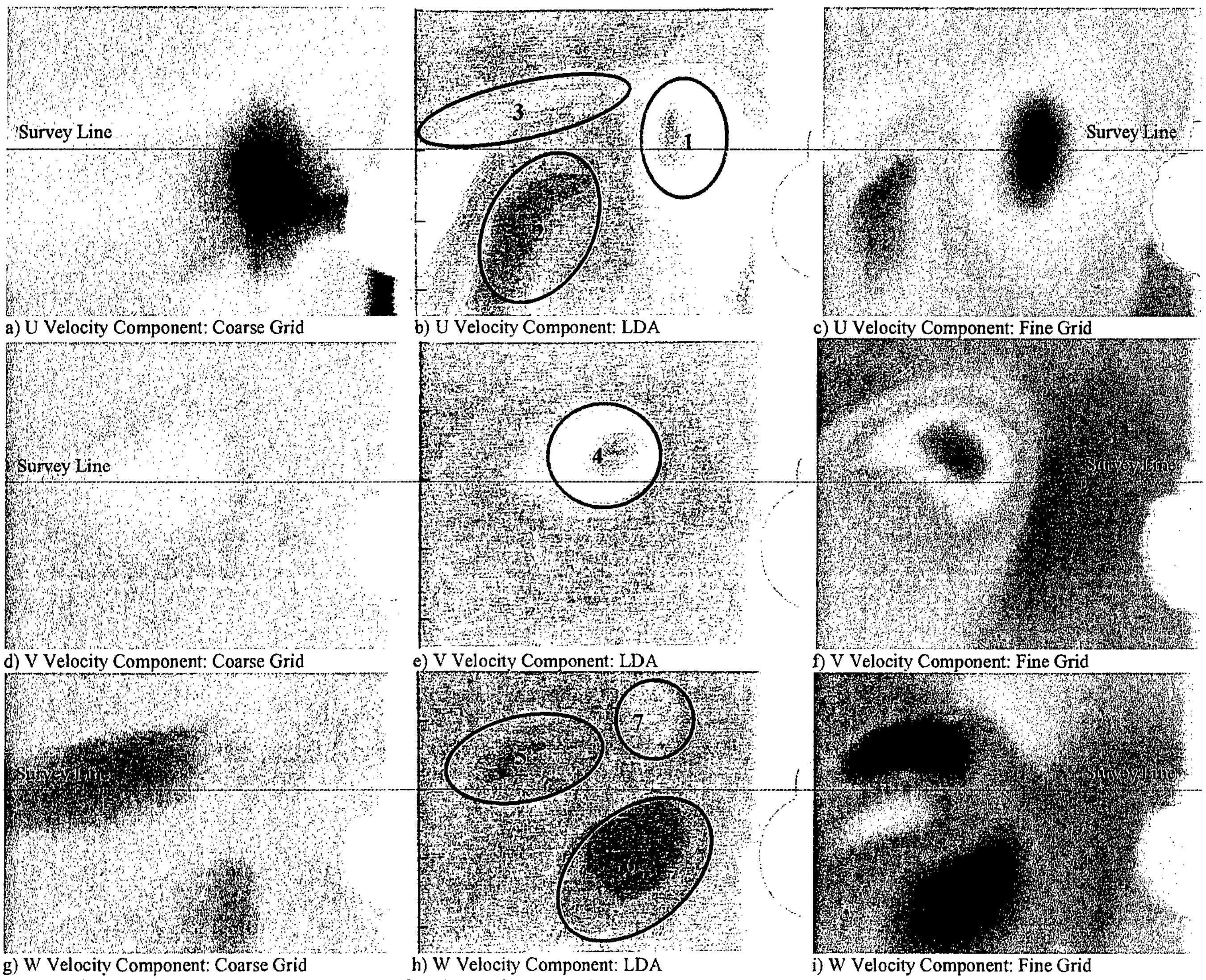
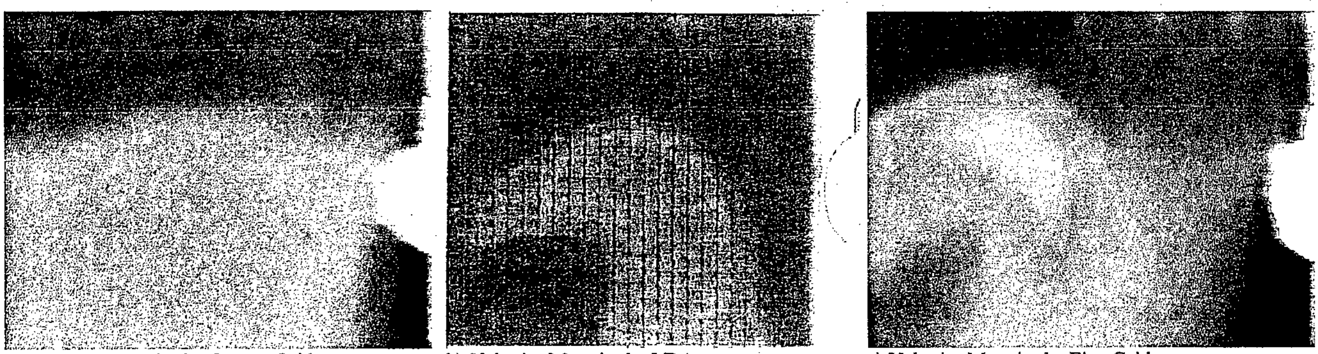


Figure 6: Coarse CFD Result, LDA Result and Fine CFD Result for U, V and W Velocity Components at y = 150.

The exhaust core region is clearly visible in figures 6d, 6e and 6f as the V velocity component is closely orientated with the exhaust direction. The fine CFD result shows a fair representation of the core region (region 4) with the predicted location offset further from the tail boom than in the LDA results. The coarse grid shows the core to be quite diffuse and indistinct.

The three main features exhibited by the LDA W velocity component (regions 5, 6, and 7) are predicted in both CFD models, again with the highest resolution given using the fine grid.

Figure 7 shows the total velocity magnitude given by the coarse CFD result, the fine CFD result and the LDA data at y=300. The preliminary CFD model shows a reasonable prediction of the exhaust flow velocity at this point but the shape of the exhaust flow is diffuse. The LDA data shows a distinct bowed region of flow that is duplicated in the fine CFD result. The fine result over estimates the exhaust velocity and predicts the plume slightly further out and higher relative to the tail boom than in the measurements.



a) Velocity Magnitude: Course Grid b) Velocity Magnitude: LDA c) Velocity Magnitude: Fine Grid

Figure 7: Course CFD Result, LDA Result and Current CFD Result for Total Velocity Magnitude at y = 300.

Figure 8 shows the velocity component and magnitude profiles along a survey line perpendicular to the tail boom in the plane y = 150 for each of the data sets, as marked in figure 6. Both the coarse and the fine grid result in a prediction that shows the general trends exhibited by the LDA data along the survey line.

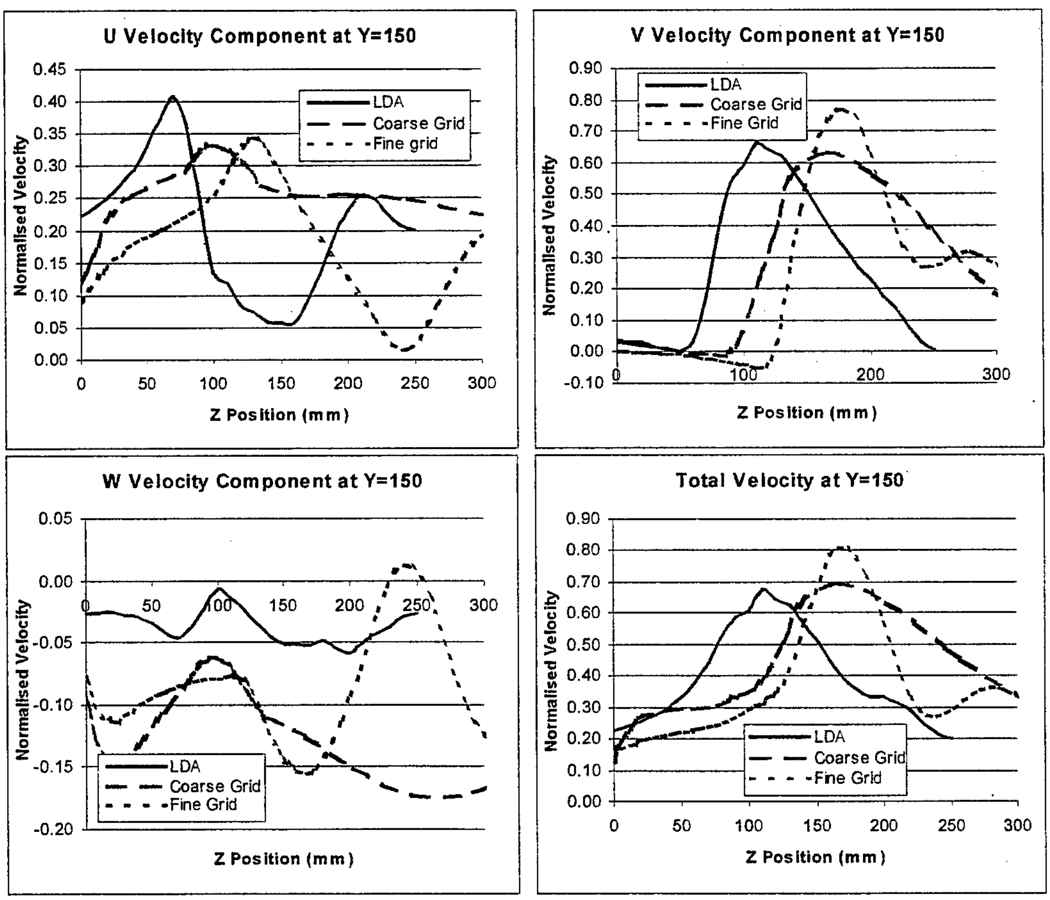


Figure 8: Velocity Components and Total Velocity Magnitude Along a Survey Line in the y = 150 Plane.

The velocity profile for the U component clearly shows the offset of the core exhaust region discussed above and the differences in the result using the coarse and the fine grid. Both CFD models predict the peak in downward velocity with the CFD showing this region to be further out relative to the boom than the measurements. The fine grid model follows the pattern exhibited by the LDA data quite closely while the coarse grid model fails to pick out the trough completely. This is partly due to the flow features being predicted lower relative to the survey line in the coarse grid result but the trough is still under estimated if the survey line is moved to compensate.

The V velocity component is roughly orientated with the exhaust flow and rises to a peak as the survey point passes through the exhaust core. Again the displacement outwards is seen in the CFD results. As the exhaust flow is roughly five times that of the downwash, the V component is dominant and hence the total velocity magnitude exhibits very similar trends and magnitudes as the V velocity component. The core velocity is predicted to be greater at this point than the measured result, showing the predicted plume is mixing and dissipating slower.

The W velocity component shows the worst correlation between predicted and measured data. Again the fine grid captures the peaks and troughs along the survey line better than the model using the coarse grid but both CFD results are greater in magnitude than the LDA data.

Figure 9 shows the U and V flow components along the axial centreline of the exhaust. The measured and predicted U and V velocity components show similar characteristics in both general trend and magnitude, though the CFD results generally lead the LDA line slightly. As has been discussed above, the predicted exhaust is flowing at a greater effective angle to the tail boom than the LDA data indicates. This means that the axial survey line passes out of the core exhaust region more quickly for the CFD results and hence the velocity components tend towards the free stream more rapidly.

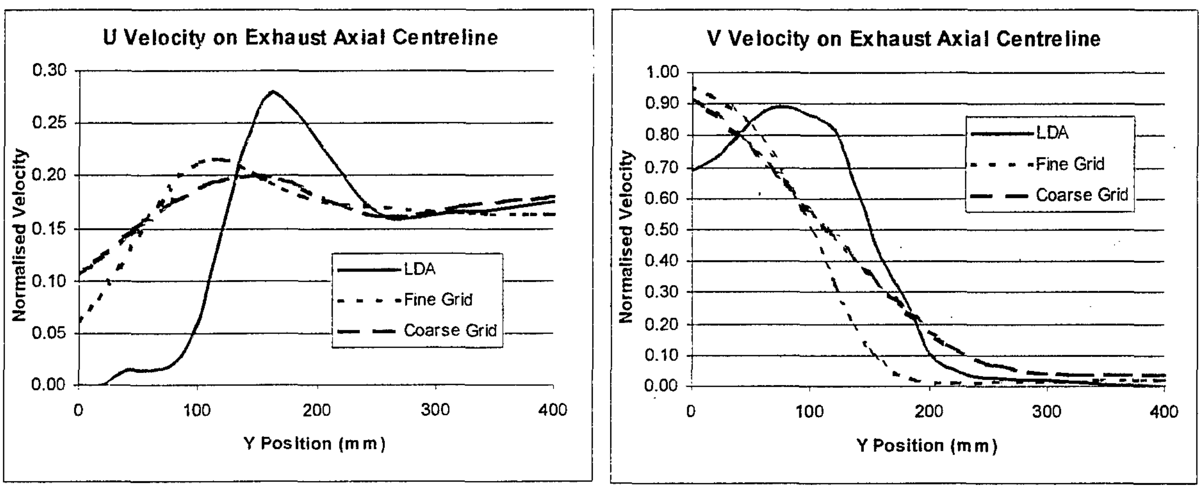


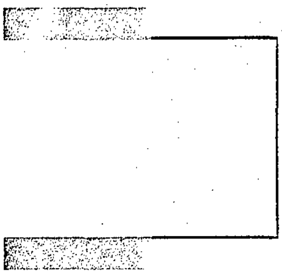
Figure 9: Velocity Components Along an Axial Survey Line.

DISCUSSION/CONCLUSION

This investigation has provided further confidence that SAPPHIRE is capable of predicting the flow field characteristics necessary to provide a fit for purpose thermal evaluation for helicopter signature analysis. While it has been shown that SAPPHIRE can predict the general flow features relating to the exhaust/downwash interaction, there are discrepancies between predicted and measured plume location and flow mixing. The solution is only as good as the boundary conditions used to define and constrain the flow field. A more accurate prediction of this case would require resolving several key features that are not available within the LDA data.

The most obvious factor is the boundary condition used for the exhaust in the CFD model. The exhaust boundary condition was set such that the outlet had a 'top hat' mass flow and a velocity vector perpendicular to the exit plane, i.e. the exhaust flow was at a constant magnitude and direction across the entire exit plane (figure 10a). In reality this is not going to be the case. The exhaust flow is piped through the model, undergoing several changes in direction and as such is unlikely to be

flowing perpendicular to the exit plane. Coupled with this, boundary layer effects within the exhaust duct will cause a velocity profile as shown in figure 10b.


a) 'Top Hat' Velocity Boundary

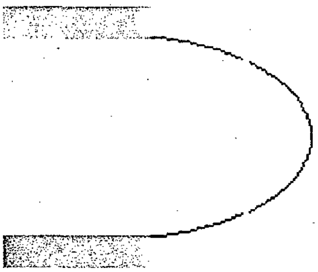
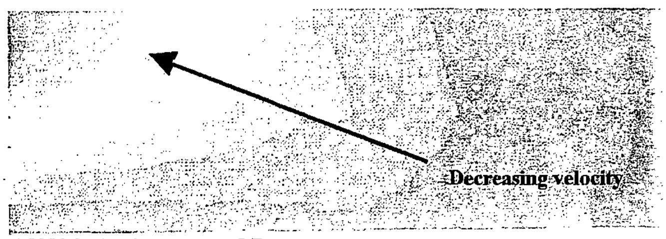

b) Profiled Velocity Boundary

Figure 10: Difference between 'Top Hat' and 'Profiled' Velocity Boundaries.

The CFD calculation domain was bounded by free boundaries while the wind tunnel used a closed section design. The helicopter model caused a significant blockage to the free stream flow giving a varied flow profile above the model. The CFD inlet boundary above the helicopter was again given a top hat value based on the wind tunnel velocity. The U velocity component plots at the effective rotor plane for the LDA and the fine grid model are shown in figure 11. This clearly shows the reduced downwash in both the experimental and computational results in this region. However, the LDA data shows a significantly higher decrease. The reduction in free stream flow would lessen the effect of the 'downwash' on the exhaust, particularly close to the fuselage and exhaust exit. This, together with the uncertainty of the flow conditions at the nozzle exit, is the probable cause of the discrepancy in exhaust flow position.



a) U Velocity Component: LDA

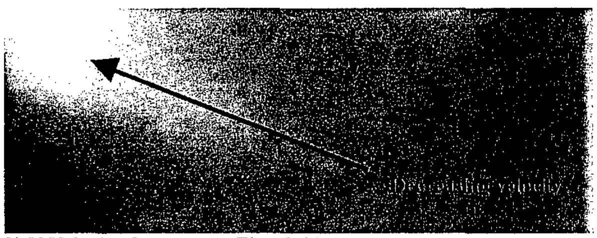

b) U Velocity Component: Fine Grid

Figure 11: U Velocity Component in Rotor Plane.

The slower mixing and attenuation of the exhaust in the predicted results than the LDA data may also indicate that the turbulence boundary conditions and predicted intensity used in the CFD calculations were lower than they were in reality. The use of a different turbulence model may also improve the correlation between the results and further work is being undertaken to investigate this factor.

Finally the position of the survey lines and planes for the CFD data set may not correspond precisely with those used for the LDA measurements. The reference point for the LDA measurements was a point just aft of the exhaust and is not precisely known.

ACKNOWLEDGMENTS

This work has been funded under UK MOD Applied Research Package 12 and carried out by Dstl Farnborough.

The 'Rotor Downwash Study', through which the wind tunnel measurements were generated, was jointly funded by UK MOD and Rolls Royce Plc. The Authors would like to thank Rolls Royce for the use of this experimental data (copyright Rolls-Royce Plc 1998).

The Authors would like to thank S&C Thermofluids for their support and help with SAPPHIRE.

REFERENCES

[1] Rooks, Steve. Smith, Anthony G., 1997, "Flow Field and IR Signature Modelling for Ground Targets", *Proceedings of the Eighth Annual Ground Target Modelling and Validation Conference,* Houghton, MI.

[2] Lavery, N. Brown, S. Spittle, J. Rooks. S. Hayward, L. Smith, A. 2000, "A Coupled CFD-Thermal Solver Routine for Detailed Platform IR Prediction", *Proceedings of the Eleventh Annual Ground Target Modelling and Validation Conference,* Houghton, MI.

[3] Lavery, N. Brown. S. Spittle, J. Rooks, S. Hayward, L. Smith, A. 2001, "Improved Modelling Capabilities of PHARO – A Coupled CFD-Thermal Solver for Detailed Platform IR Prediction", *Proceedings of the Twelfth Annual Ground Target Modelling and Validation Conference,* Houghton, MI.

[4] Newman, S. Rooks, S. Bull, C. Green, J. Taylor, K. Smith. A. 2001, "Validation of SAPPHIRE Heat Transfer Predictions", *Proceedings of the Twelfth Annual Ground Target Modelling and Validation Conference,* Houghton, MI.

[5] Research and Technology Organisation (RTO), 1999, "A feasibility Study of Collaborative Multi-Facility Windtunnel Testing for CFD Validation" RTO TR-27.

[6] Launder, B.E. Spalding, D. B. 1972, "Mathematical Models of Turbulence" *Academic Press.*